EXPERIMENTAL AND 2D NUMERICAL INVESTIGATION OF THE FLOW IN AN INDUCER WITH STRAIGHT HYDROFOIL CASCADE

Liviu ANTON, Prof.*
Department of Hydraulic Machinery
"Politehnica" University of Timisoara

Adrian STUPARU, Assist. Prof.
Department of Hydraulic Machinery
"Politehnica" University of Timisoara

Sebastian MUNTEAN, Senior Researcher
Centre of Advanced Research in Engineering Sciences
Romanian Academy - Timisoara Branch

Alexandru BAYA, Prof.
Department of Hydraulic Machinery
"Politehnica" University of Timisoara

*Corresponding author: Bv Mihai Viteazu 1, 300223, Timisoara, Romania
Tel.: (+40) 256 403692, Fax: (+40) 256 403700, Email: liviu.anton@tiriacauto.ro

ABSTRACT

This paper presents the numerical investigation of the 2D flow in a hydrofoil cascade of an inducer by using commercial code FLUENT 6.3. First the investigated inducer is described, then the equations that govern the flow and the boundary conditions imposed and in the end the results of the flow simulation are compared with the experimental results and a set of conclusions is presented.

KEYWORDS

inducer, numerical investigation, turbulent flow, velocity distribution analyse

NOMENCLATURE

t	[s]	time
p	[Pa]	pressure
V	[m/s]	absolute velocity
w	[m/s]	relative velocity
Q	[m^3/s]	flow rate
H	[m]	pumping head
β_s	[º]	stager angle
β_o	[º]	inflow angle
c_p	[-]	pressure coefficient
g	[m/s^2]	gravity
ρ	[kg/m^3]	density
k	[m^2/s^2]	turbulent kinetic energy

1. INTRODUCTION

The inducer is an axial impeller which is mounted on the same shaft with the impeller of the pump. By producing a supplementary specific energy on the inlet of the impeller of the pump, because of the rising of the pressure over the vaporising pressure, it prevents or reduces the cavitation. In the case of using an inducer with a pumping head of H_{ind}, the level of the minimum pressure from the impeller of the pump is rising in correspondence with this pumping head.

From figure 1, one may observe that the minimum pressure in a point *M* rises from $p_{\min}$ to $p_{\min c}$ if an inducer is used. That is why is important that the pumping head produced by the inducer, H_{ind}, to be as high as possible. This is realised by the optimum design of the geometry of the inducer.

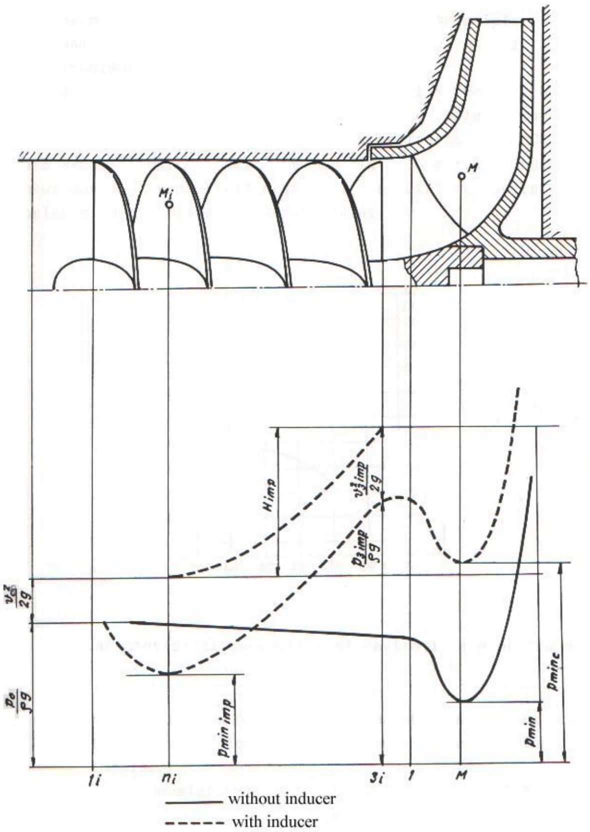

Figure 1. The effect of an inducer mounted on the shaft of a pump

2. THEORETICAL CONSIDERATION

The inducer is used usually for the pumps with severe suction condition and it has a “sacrificial role”. It will be replaced after a certain operating time and that is why the technology for its execution has to be simple and economical.

For the classical inducers the hydrofoil for the blades consists of straight hydrofoils with constant thickness like in figure 2. The blades of these inducers are materialised by a ruled surface generated by a straight line positioned on a circular propeller with constant slope and it is always perpendicular on the axis of the impeller.

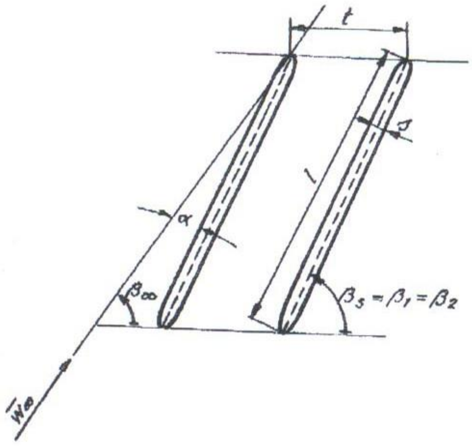

Figure 2. Straight hydrofoil of constant thickness for blade of the inducer

In conclusion, the usual inducer is much alike a screw.

Using the design equations for the turbo machinery blades, it is obtaining o slight curved shape for the camber line of the hydrofoil with better hydrodynamic conditions for the flow. This solution is difficult to realise from the technological point of view.[1]

Tacking into consideration the influence of the angle β_2 on the theoretical pumping head, as described by the equation 1:

$$H_t = \frac{u_2}{g(1+p)}\left(u_2 - \frac{v_{m2}}{tg\beta_2}\right) \tag{1}$$

which shows the raising of the pumping head with increasing of the angle β_2, in this paper it is proposed a straight cascade. This network has the stagger angle β_s equal with the angle β_2 of the curved hydrofoil cascade resulted from the designing calculation. This type of the inducer will operate at nominal flow rate with large incidence angle on the inlet and will present unsatisfactory cavitational characteristics. It is recommended that this type of inducer to be used to flow rates higher than the nominal flow rate, where the cavitational coefficient $NPSH_r$ has higher values.

Using the nominal parameters of a centrifugal pump used in chemic industry, with a suction diameter $D_0 = 100mm$ and the discharge diameter $D_2 = 200mm$, and the following characteristics $Q = 100m^3/h$, $H = 50m$ and $n = 2900rot/\min$, an inducer was designed and manufactured with the exterior diameter $D = 100mm$ and the shaft diameter $d = 30mm$.

This inducer, figure 3, has the hydrofoil cascade formed by straight hydrofoil with a rounded leading and trailing edge like in figure 2, has two blades and was numerical and experimental investigated.

Figure 3. The inducer investigated experimental and numerical

3. COMPUTATIONAL DOMAIN, EQUATIONS AND BOUNDARY CONDITIONS

The computational domain, figure 4, was generated using the pre-processor GAMBIT from FLUENT, based on the existing geometry. The geometrical characteristic of the hydrofoil corresponding to the middle radius of the blade and the investigated operating condition are given in table 1:

Table 1. Geometrical characteristic and operating conditions of the investigated hydrofoil

Q [m³/h]	**n [rot/min]**	**D [mm]**	**d [mm]**	**s [mm]**	**β_s [º]**	**β_0 [º]**	**t/l [-]**
100	2900	100	30	3	36.5	20.52	0.804

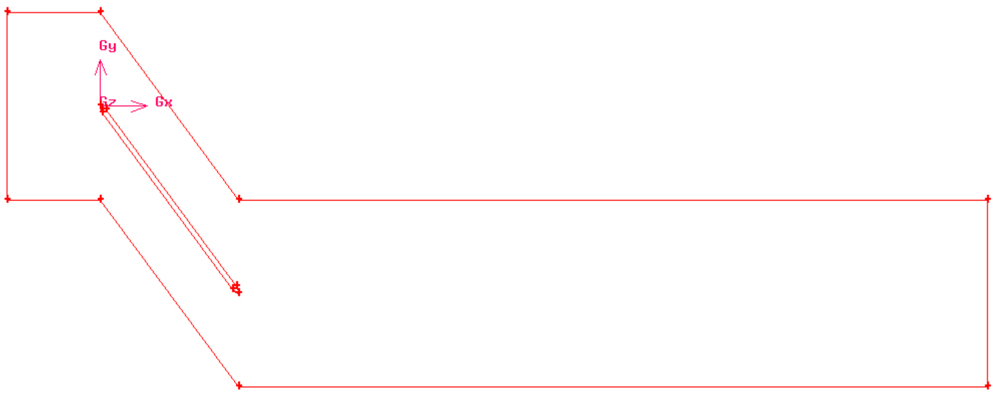

Figure 4. Computational domain of the inducer

The generated mesh for the computational domain is structured and has 60,000 cells, figure 5. A boundary layer was attached to the hydrofoil in order to be able to compute the flow near a solid wall.

The periodic boundaries of the domain are positioned at a distance, regarding the chord, equal with the space of the cascade, while the inlet is positioned at a distance equal with half of the space of the cascade and the outlet at a distance equal with four times the space of the cascade.

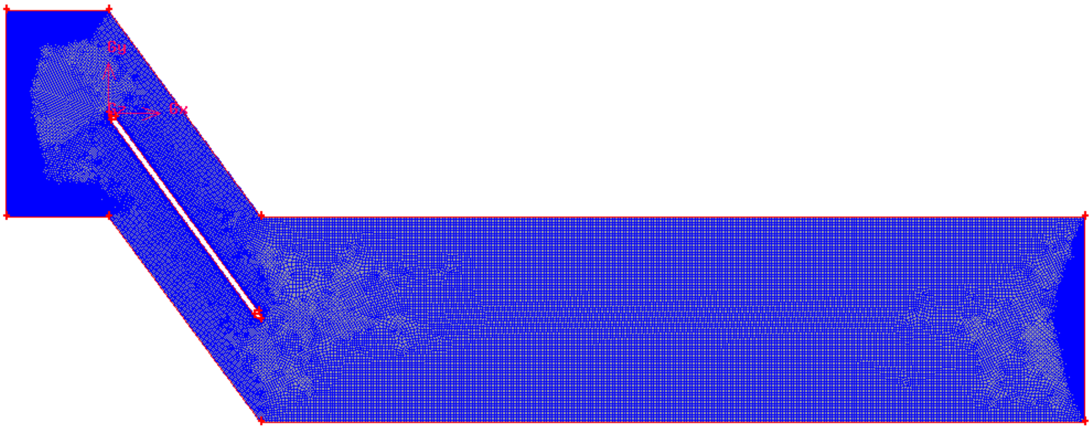

Figure 5. Mesh generated on the 2D computational domain of the inducer

For the flow analysis presented in this paper we consider a 2D turbulent flow model. A steady relative 2D flow is computed in the computational domain,

$$\nabla \cdot \vec{V} = 0 \qquad (2)$$

$$\frac{d\left(\rho \vec{V}\right)}{dt} = \rho g - \nabla p + \mu \Delta \vec{V} \qquad (3)$$

The numerical solution of flow equations (2) and (3) is obtained with the expert code FLUENT 6.3, [3], using a Reynolds-averaged Navier-Stokes (RANS) solver. As a result, the viscosity coefficient is written as a sum of molecular viscosity μ and turbulent viscosity μ_T, and the last term in the right-hand-side of (2) becomes $\nabla \cdot \left[(\mu + \mu_T) \nabla \vec{V} \right]$. For steady, absolute flow the left hand side of (2) reduces $\nabla \cdot \left(\rho \vec{V} \vec{V} \right)$

The turbulent viscosity is computed using the RSM model which is characterised by the following equations:

$$\begin{aligned}
&\underbrace{\frac{\partial}{\partial t}\left(\rho \overline{u_i' u_j'}\right)}_{\text{local time derivative}} + \underbrace{\frac{\partial}{\partial x_k}\left(\rho u_k \overline{u_i' u_j'}\right)}_{C_{ij} \equiv \text{ convection}} = \underbrace{-\frac{\partial}{\partial x_k}\left(\rho \overline{u_i' u_j' u_k'} + \overline{p\left(\delta_{kj} u_i' + \delta_{ik} u_j'\right)}\right)}_{D_{T,ij} \equiv \text{ turbulent diffusion}} + \\
&+ \underbrace{\frac{\partial}{\partial x_k}\left[\mu \frac{\partial}{\partial x_k}\left(\overline{u_i' u_j'}\right)\right]}_{D_{L,ij} \equiv \text{ molecular diffusion}} \underbrace{-\rho\left(\overline{u_i' u_j'} \frac{\partial u_j}{\partial x_k} + \overline{u_j' u_k'} \frac{\partial u_i}{\partial x_k}\right)}_{P_{ij} \equiv \text{ stress production}} \underbrace{-\rho \beta \left(g_i \overline{u_j' \theta} + g_j \overline{u_i' \theta}\right)}_{G_{ij} \equiv \text{ buoyancy production}} + \qquad (4) \\
&\underbrace{+ \overline{p\left(\frac{\partial u_i'}{\partial x_j} + \frac{\partial u_j'}{\partial x_i}\right)}}_{\Phi_{ij} \equiv \text{ pressure strain}} \underbrace{- 2\mu \overline{\left(\frac{\partial u_i'}{\partial x_k} \frac{\partial u_j'}{\partial x_k}\right)}}_{\varepsilon_{ij} \equiv \text{ dissipation}} \underbrace{-2\rho \Omega_k \left(\overline{u_j' u_m'} \varepsilon_{ikm} + \overline{u_i' u_m'} \varepsilon_{jkm}\right)}_{F_{ij} \equiv \text{ production by system rotation}} + \underbrace{S_{user}}_{\text{user-defined source term}}
\end{aligned}$$

We imposed on the inlet section the two components of the velocity, corresponding to the prescribed flow rate and flow angle, together with the turbulence parameters.

$$w_x = \frac{Q}{S_{IN}} \tag{5}$$

$$w_y = \frac{w_x}{tg\beta_0} \tag{6}$$

On the outlet section of the computational domain a radial equilibrium condition is chosen.

On the periodic surfaces of the domain the periodicity of the velocity, pressure and turbulence parameters were imposed.

The remaining boundary conditions for the domain correspond to zero relative velocity on the suction side and pressure side of the hydrofoil.

4. NUMERICAL AND EXPERIMENTAL RESULTS

The pressure coefficient is calculated with the following relation:

$$c_p = \frac{p - p_{IN}}{\frac{\rho}{2} w_{IN}^2} \tag{7}$$

The test rig for the experimental investigation of the flow in an inducer is presented in figure 6.

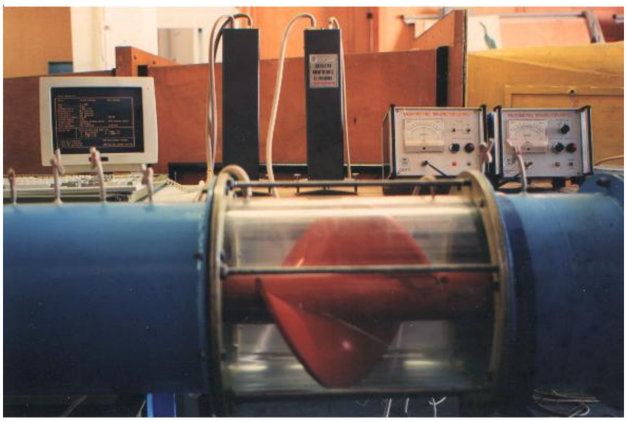

Figure 6. Test rig for pressure measurement on the blade of an inducer

This is a wind tunnel in open circuit with the radius of the measuring section of 300 mm. The inducer model has drained blades and allowed, through two scan valves sealed with magnetic fluid, the measurement and the calculation of the pressure distribution along the blades [2]. The inducer model tested in air was realised at the scale

$$\frac{D_{water}}{D_{air}} = \frac{1}{3}$$

with the respect of the laws of similitude.

For the validation of the numerical result a comparison between the pressure coefficient obtained from the numerical investigation and from experimental investigation is made, figure 7.

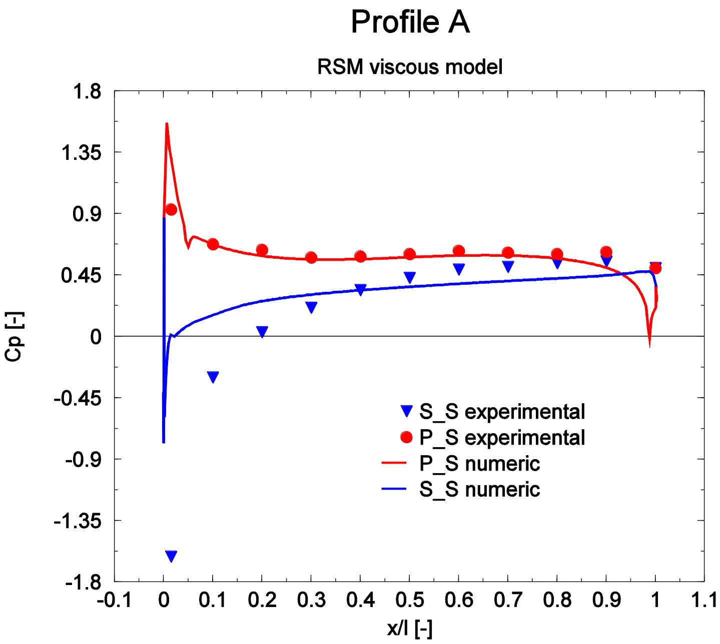

Figure 7. Pressure coefficient distribution along the blade of the inducer

It can be observed that the pressure coefficient distribution along the pressure side of the blade is almost identical for the numerical simulation of the flow and the experimental investigation. For the suction side of the blade the distribution of the pressure coefficient is not alike for the numerical and experimental results because the detachment of the fluid from the hydrofoil.

In figure 8 the pattern of the flow inside the inducer is presented with the help of the distribution of the stream lines. It can be observed that on the suction side of the blade a detachment of the flow is present. This phenomenon appears due to the value of the flow angle which leads to the fact that the incidence point of the flow to be situated on the pressure side and not on the leading edge of the hydrofoil.

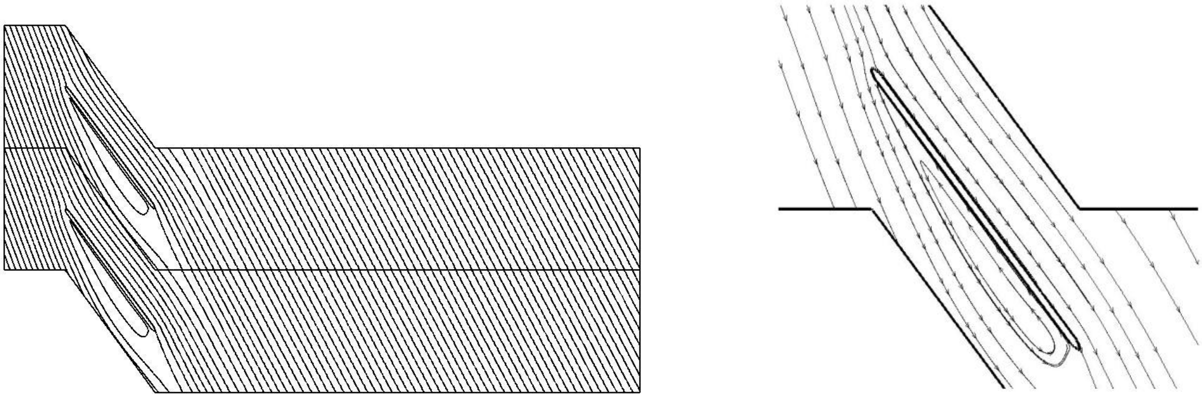

Figure 8. Stream lines distribution for the flow inside the inducer

The detachment of the fluid from the suction side of the blade of the inducer can be observed also from the velocity distribution presented in figure 9.

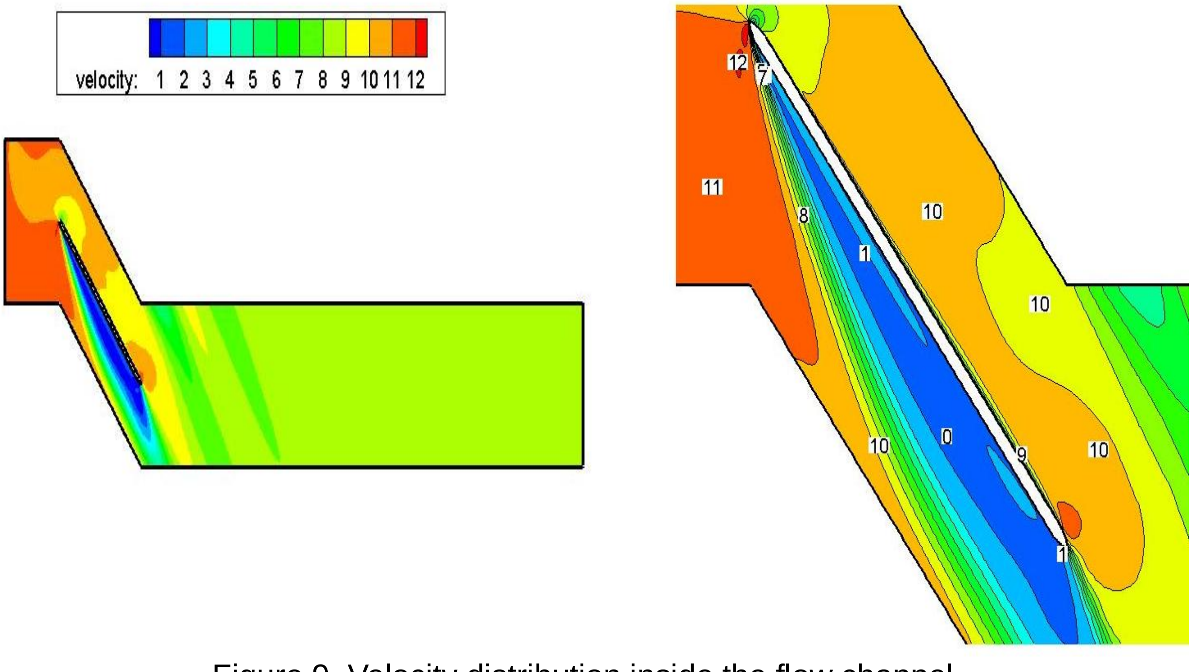

Figure 9. Velocity distribution inside the flow channel

5. CONCLUSIONS

From the comparison of the numerical data with the experimental data regarding the pressure coefficient distribution along the blade of the inducer it results a good agreement. This proves that the boundary conditions and the RSM turbulence model were adequate chosen. The numerical investigation of the flow inside the inducer predicts the presence of a dead zone of the flow on the suction side of the hydrofoil. This leads to the conclusion that the straight hydrofoil is not suited for the construction of the blade of the inducer.

Further investigation will be performed regarding the use of a bended hydrofoil for the construction of the blade of the inducer.

ACKNOWLEDGMENTS

The authors acknowledge the support from the National University Research Council grants. All numerical computations have been performed at the Numerical Simulation and Parallel Computing Laboratory of the “Politehnica” University of Timisoara, National Centre for Engineering of Systems with Complex Fluids.

REFERENCES

1. Anton, L.E., Milos, T., *Centrifugal pumps with inducer*, Ed. Orizonturi universitare, Timisoara,1998
2. Anton, L.E., *Determination of pressure distribution on the blades of an inducer*, XVII IAHR Symposium, Beijing, China,1994
3. Stuparu, A., Muntean, S., Balint, D., Baya, A., Anton, L. E., *Numerical investigation of the influence of the suction-elbow on the inlet flow field of a storage pump*, Scientific Bulletin of the „Politehnica” University Of Timisoara, Transactions on Mechanics, Tom 51(65),Fascicola 3, 2006, ISSN 1224-6077, pp. 31…36
4. Stuparu, A., Muntean, S., Balint, D., Baya, A., Anton, L. E., *Numerical Analysis of Pump Hydrodynamics at Constant Speed*, Scientific Bulletin of the „Politehnica” University Of Timisoara, Transactions on Mechanics, Tom 52(66),Fascicola 3, 2007, ISSN 1224-6077
5. Akhras, A., El Hajem, M., Morel, R., Champagne, J.Y., *The Internal Flow Investigation of a Centrifugal Pump*, Proceeding of Flucome, Sherbrooke, Canada, 2000
6. Fluent Inc., *FLUENT 6.3 User’s Guide*, Fluent Incorporated, 2001.
7. Fluent Inc., *Gambit 2.2.30 User’s Guide*, Fluent Incorporated, 2001.
8. Hofmann, M., Stoffel, B., *Experimental And Numerical Studies On A Centrifugal Pump With 2d-Curved Blades In Cavitating Condition*, Proceeding of Fourth International Symposium on Cavitation CAV2001, Pasadena, California, SUA
9. Muntean, S., *Numerical methods for the analysis of the 3D flow in Francis turbine runners*, PhD Thesis Politehnica University of Timisoara, Timisoara, 2002.
10. Sallaberger, M., Sebestyen, A., Mannschreck, E., Pinkas, W., *Modern Pump Impeller Design With Consideration Of Non-Uniform Inlet Flow Fields*, Proceedings of IAHR, Graz, 1999

11. Tamm, A., Ludwig, G., Stoffel, B., Numerical, *Experimental And Theoretical Analysis Of The Individual Efficiencies of a Centrifugal Pump*, Proceedings of ASME FEDSM´01 2001 ASME Fluids Engineering Division Summer Meeting New Orleans, Louisiana, May 29 – June 1, 2001
12. Tamm, A., Bråten, A., Stoffel, B., Ludwig, G., *Analysis Of A Standard Pump In Reverse Operation Using CFD*, Hydraulic Machinery and Systems, 20th IAHR Symposyum, 2001
13. Van Esch, B. P. M., *Simulation of three-dimensional unsteady flow in hydraulic pumps*, Thesis University of Twente, Enschede, 1997